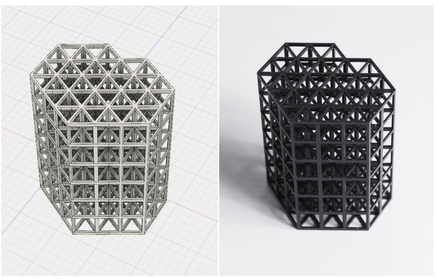
The additive manufacturing process

Written by Ben Redwood

Learn about the general additive manufacturing process from design to final part for those who have never 3D printed before.

Introduction

Additive manufacturing (sometimes referred to as rapid prototyping or 3D printing) is a method of manufacture where layers of a material are built up to create a solid object. While there are many different 3D printing technologies this article will focus on the general process from design to final part. Whether the final part is a quick prototype or a final functional part the general process does not change.



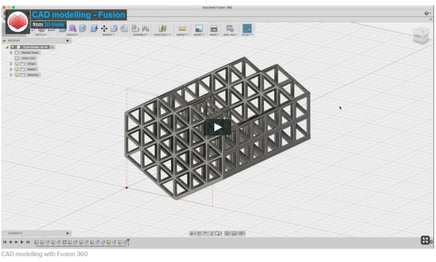
From initial CAD design to 3D printed part the additive manufacturing follows a general series of steps

Additive manufacturing process

1. CAD

Producing a digital model is the first step in the additive manufacturing process. The most common method for producing a digital model is computer aided design (CAD). There are a large range of free and professional CAD programs that are compatible with additive manufacture. Reverse engineering can also be used to generate a digital model via 3D scanning.

There are several design considerations that must be evaluated when designing for additive manufacturing. These generally focus on feature geometry limitations and support or escape hole requirements and vary by technology.

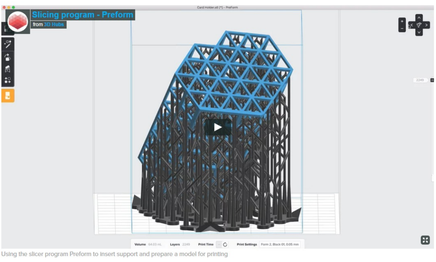


CAD modelling with Fusion 360

2. STL conversion and file manipulation

A critical stage in the additive manufacturing process that varies from traditional manufacturing methodology is the requirement to convert a CAD model into an STL (stereolithography) file. STL uses triangles (polygons) to describe the surfaces of an object. A guide on how to convert a CAD model to an STL file can be found here. There are several model limitations that should be considered before converting a model to an STL file including physical size, watertightness and polygon count.

Once a STL file has been generated the file is imported into a slicer program. This program takes the STL file and converts it into G-code. G-code is a numerical control (NC) programming language. It is used in computer-aided manufacturing (CAM) to control automated machine tools (including CNC machines and 3D printers). The slicer program also allows the designer to customise the build parameters including support, layer height, and part orientation.

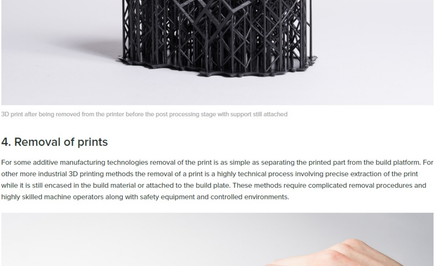


Using the slicer program Preform to insert support and prepare a model for printing

3. Printing

3D printing machines often comprise of many small and intricate parts so correct maintenance and calibration is critical to produce accurate prints. At this stage the print material is also loaded into the printer. The raw materials used in additive manufacturing often have a limited shelf life and require careful handling. While some processes offer the ability to recycle excess build material, repeated reuse can result in a reduction in material properties if not replaced regularly.

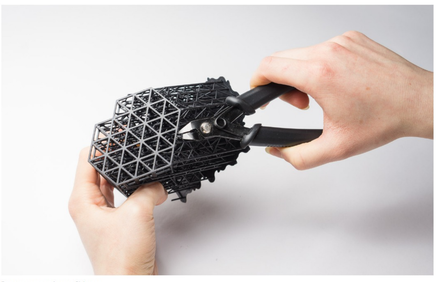
Most additive manufacturing machines do not need to be monitored after the print has begun. The machine will follow an automated process and issues generally only arise when the machine runs out of material or there is an error in the software. A explanation on how each of the different additive manufacturing printers produce parts can be found here.



3D print after being removed from the printer before the post processing stage with support still attached

4. Removal of prints

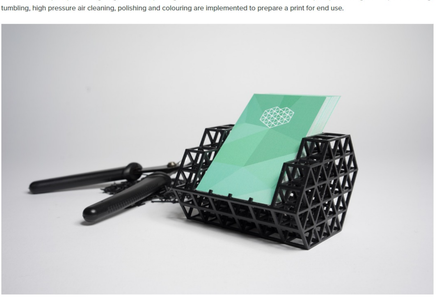
For some additive manufacturing technologies removal of the print is as simple as separating the printed part from the build platform. For other more industrial 3D printing methods the removal of a print is a highly technical process involving precise extraction of the print while it is still encased in the build material or attached to the build plate. These methods require complicated removal procedures and highly skilled machine operators along with safety equipment and controlled environments.



Removing support from an SLA print

5. Post processing

Post processing procedures again vary by printer technology. SLA requires a component to cure under UV before handling, metal parts often need to be stress relieved in an oven while FDM parts can be handled right away. For technologies that utilize support, this is also removed at the post processing stage. Most 3D printing materials are able to be sanded and other post processing techniques including tumbling, high pressure air cleaning, polishing and colouring are implemented to prepare a print for end use.



A business card holder made with additive manufacturing

Written by

Ben Redwood
Mechanical engineer working at 3D Hubs